

# ECE322L - Lab 1

## *Circuit Simulation*

Roger Holten, David Kirby, Landon Schmucker

---

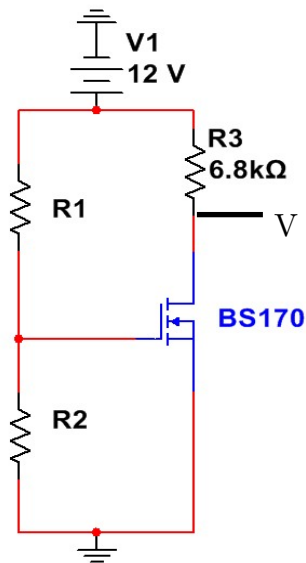
### Goal

Explore and gain understanding of PSpice and Multisim software.

### Software needed

- PSpice and Multisim

### The Lab



*Figure 1: Circuit Diagram*

### Part 1 – Hand Calculations

Let  $K'n = 1.825 \frac{mA}{V^2}$ ,  $\left(\frac{W}{L}\right) = 1$ ,  $V_{TN} = 1V$ , and  $R_1 + R_2 = 100k\Omega$ . Bias the transistor such that half the supply voltage is across the transistor.

1. Find  $I_{DQ}$  such that half the supply voltage is across the transistor.
2. Find  $R_1$  and  $R_2$  through hand calculations.

3. Solve for  $V_{DSQ}$  and  $V_{GSQ}$ .

	Hand Calc	PSpice	Multisim
$I_{DQ}$	0.8824mA	0.876mA	0.876mA
$V_{DSQ}$	6V	6.04V	6.04V
$V_{GSQ}$	1.9833V	1.98V	1.98V
$R_1$	83.472k $\Omega$	N/A	N/A
$R_2$	16.528k $\Omega$	N/A	N/A

LAB 01 ECE 322 L 29-Jan

Hand Calculations:

$$K'_n = 1.825 \frac{\text{mA}}{\text{V}^2} \quad \frac{W}{L} = 1 \quad V_{TN} = 1\text{V}$$

$$R_1 + R_2 = 100\text{k}\Omega \quad R_3 = 6.8\text{k}\Omega \quad V_{DD} = 12\text{V}$$

$$I_D = \frac{1}{2} K'_n \left(\frac{W}{L}\right) (V_{GS} - V_{TN})^2 \quad V_{GS} = V_{DD} \frac{R_2}{R_1 + R_2}$$

$$V_{DS} = V_{DD} - R_D I_D \quad V_{GS} = 12\text{V} \left(\frac{R_2}{100\text{k}\Omega}\right)$$

$$6\text{V} = 12\text{V} - 6.8\text{k}\Omega I_D$$

$$6.8\text{k}\Omega I_D = 6\text{V}$$

$$I_{DQ} = 0.8824\text{mA}$$

$$0.8824\text{mA} = \frac{1}{2} \left(1.825 \frac{\text{mA}}{\text{V}^2}\right) (1) (V_{GS} - 1\text{V})^2$$

$$V_{GSQ} = 1.9833\text{V}$$

$$R_2 = 16.528\text{k}\Omega \quad R_1 = 83.472\text{k}\Omega$$

$$V_{DSQ} = V_{DD} - R_D I_D = 6\text{V}$$

## Part 2 – Modeling the Circuit in PSpice

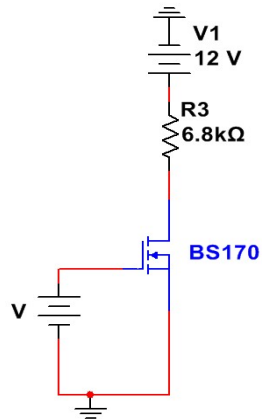


Figure 2: PSpice Circuit Diagram

1. Create the circuit in PSpice

```
V1 1 0 12
*Gate Voltage, we will use this to sweep for VGSQ
Vg 2 0 1.98

*Drain Resistor < + node> < -node > <Resistance>
Rd 1 3 6.8k

*<label> <drain> <gate> <source> <bulk> <.model name>
M1 3 2 0 0 ntype
.model ntype nmos vto=1 kp=1.825m l = 1.0u w = 1.0u

*.dc VG 0 4 0.01

.probe
*operating point
.probe
.op
.end
```

Figure 3: PSpice Netlist

2. Provide a graph with  $V_D$  as a function of  $V_G$  in your report.

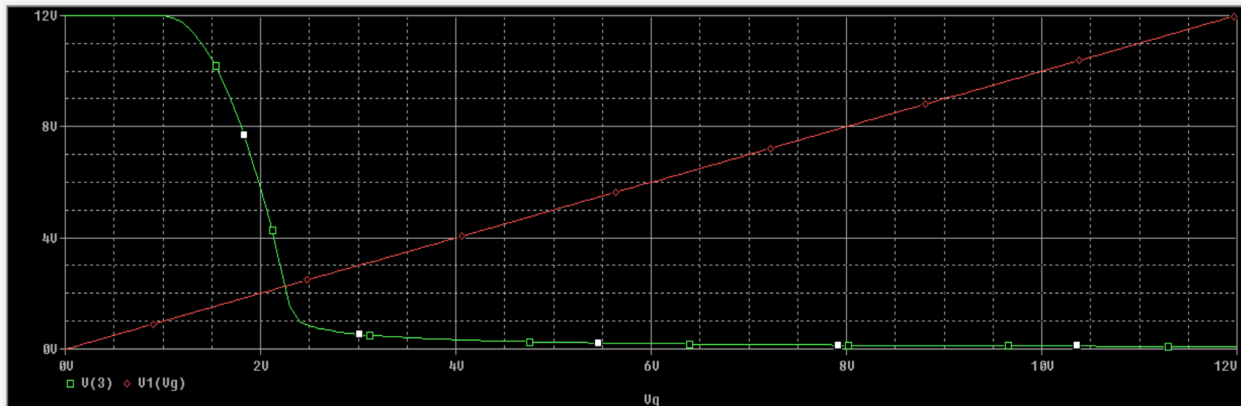


Figure 4: Graph of  $V_D$  vs  $V_G$

```

**** MOSFETS

NAME      M1
MODEL     ntype
ID        8.76E-04
VGS       1.98E+00
VDS       6.04E+00
VBS       0.00E+00
VTH       1.00E+00
VDSAT     9.80E-01

```

Figure 5: Quiescent Values with Output File

### Part 3 - Multisim

1. Create the circuit shown in Figure 2. In Multisim we place probes which can help with quick analysis and setting up simulations.

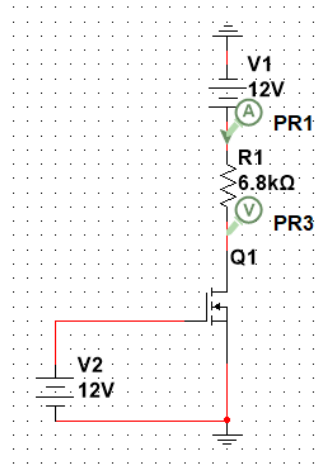


Figure 6: Multisim Circuit Diagram

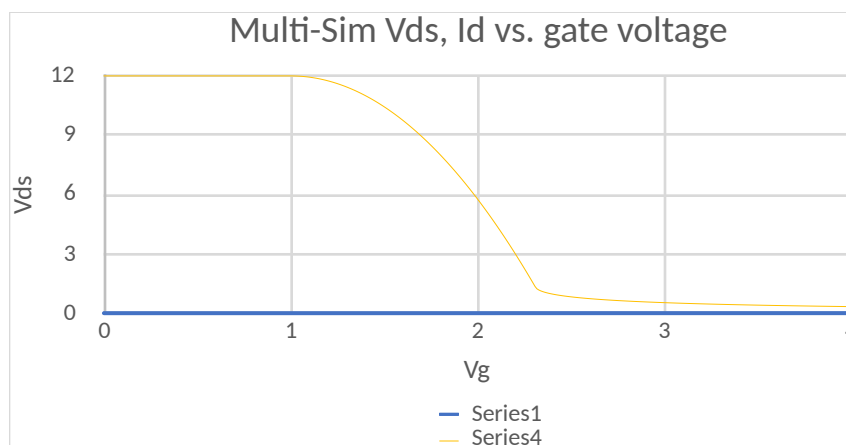


Figure 7: Graph of DC Sweep

2. Compare the differences of the quiescent values obtained through PSpice and hand calculations. What could cause the differences?

The differences between quiescent values of PSpice and hand calculations is minimal and most likely just due to rounding in the PSpice software. Another possible issue is in the PSpice we only did steps of 0.01V for the DC Sweep which would cause the small rounding error.

3. Which simulation software do you prefer? Why?

We personally prefer PSpice due to its ability to work with netlist and visual circuit creation. It also tends to be a little easier to do multiple simulations at once as you can just add them to the end of the netlist. It also is easier to edit the components since you can quickly change the netlist.